

Rough and finish threading at different speeds and with different tools

If you've ever tried to use two different spindles speeds when threading a workpiece on a turning center, you've probably been very disappointed. For whatever reason, most turning center controls (including Fanuc) cannot correctly adjust the entry point for the threading tool after a spindle speed change. While the threading tool will cut the appropriate pitch, the tool will cross-thread to some extent – much like what happens when a multiple-start thread is machined. Since the finishing cut will be made at full depth, this will, of course, damage the workpiece and in most cases, will cause the insert to break.

By the way, this is the reason why you are required to thread in the rpm mode (G97 on Fanuc controls). Again, the control cannot perfectly synchronize the entry point for the threading tool after even minor rpm changes that would occur during threading – because of slight diameter changes – in the constant surface speed mode (G96 on Fanuc controls).

For this reason, most turning center programmers simply use the same spindle speed for chasing the entire thread. But the same advantages that can be attained by roughing and finishing with different spindle speeds for other machining operations can be achieved with threading operations. These advantages include improved surface finish, longer tool life, longer periods between offset adjustments, and better overall workpiece quality. And for large production volumes, it may be quite

advantageous to thread with two separate tools – one for roughing and another for finishing.

Given the need for a precise entry point for the threading tool, most programmers don't even think about using two separate speeds for threading – let alone using two threading tools. In this article, we provide you with a procedure to rough and finish thread with two separate spindle speeds and even to do so with two separate tools. While it does require some effort during setup, it's not at all difficult to do. Additionally, we'll show a series of slash-coded commands that can be used during setup (for trial machining) to synchronize the entry point after changing speeds and/or changing threading tools.

What the setup person must do

We first describe a rather crude procedure. The goal here is to help you understand what must be done to synchronize the entry point for the finishing pass/es – whether you're simply changing speed for finishing – or whether you're using two separate threading tools. We'll be showing a refined method a little later.

For our first example, we'll say a 3 inch diameter, 16 threads per inch (0.0625 pitch) external national standard (60 degree vee) thread must be machined. This thread is 1.0 long. We'll be using one tool for both roughing and finishing. We'll rough at 500 rpm (about 400 sfm) and finish at 650 rpm (about 500 sfm). Keep in mind that this method will work for any thread diameter and pitch – as well as for internal threads and external threads. And a little later, we'll expand it to include roughing and finishing with two separate threading tools – and we'll minimize the effort required of the setup person.

We're also assuming that the threading tool will be placed in turret station number five and that the thread is at the right-most end of the workpiece (starting at Z0).

The first task is to create a test program to help the setup person determine the amount of deviation between the entry point at the roughing speed (500 rpm in our case) and the finishing speed (600 rpm in our case). Again, our method will be a little crude for now, but rest assured that we'll improve upon it. Here is the test program:

```
O1000 (Program number)
N005 T0505 M41 (Index to threading
  tool, select low spindle range)
N010 G97 S500 M03 (Start spindle)
N015 G00 X3.2 Z0.2 (Rapid to approach
  position)
N020 G92 X2.998 Z-0.95 F0.0625
  (Make one very shallow pass – just
  scratch the workpiece)
N025 G00 X7.0 Z7.0 (Rapid to tool
  change position)
N030 M30 (End of program)
```

In This Issue:

Rough and finish threading at different speeds with different tools..... 1

Subscription Information:

The Optional Stop is published quarterly by CNC Concepts, Inc. and is distributed free of charge to people downloading it from our website (www.cncci.com). Back issues are available, but a \$10.00 charge per issue will apply to any back issues not currently published on our site. Back issues 1-50 are available for a total of \$199.00.

All techniques given in this newsletter are intended to make CNC usage more efficient, easier, and/or safer. However, CNC Concepts, Inc. can accept no responsibility for the

Note that the spindle speed in line N010 is set for the roughing passes. When the setup person runs this program, the threading tool will make one very shallow pass over the thread (about 0.001 inch deep). The goal is to simply scratch the workpiece, leaving a witness mark. This pass cannot be very deep if this is an actual workpiece – going too deep may scrap the workpiece. Depending upon how accurately the setup person has determined the threading tool's X axis program zero assignment value (stored in a geometry offset on a Fanuc control), the setup person may have to run this program more than once, adjusting the tool's X axis offset and sneaking up on the workpiece.

Though it's a little off the subject at hand, note that when the setup person has adjusted the threading tool's X offset to take a 0.001 inch deep pass (again, just scratching the workpiece), they can rest assured that when the threading tool actually machines the thread to depth (later), the thread will be machined to its appropriate depth on the very first try – assuming the programmed thread depth is correct. When you think about it, this technique may actually save some setup time when machining threads that have long cutting

times, since it eliminates the need for further trial machining (consider, for example, multiple start ACME threads that may take twenty to thirty minutes to complete).

So now we have a workpiece that has a witness mark in the form of spiral scratch. Using a colored ink (commonly called bluing), the setup person must now "blue-up" the workpiece. They will also change the S word in line N010 of the program to S650, the finishing spindle speed.

At this point, the setup person will run the test program again. Since the rpm in the program has been changed, the threading tool will not "track" in the same witness mark it did when the program was run the first time.

The deviation will show up in the form of another spiral line around the workpiece. Note that this new line could be to the right (positive) side of the original line or to the left (negative) side. This polarity is important. And since the setup person has blued the workpiece, they can easily tell which line was made first – emphasizing this polarity.

It may be a little difficult to measure the deviation from one line to the other. While this measurement

doesn't have to be absolutely perfect, the more accurate this measurement, the more precise will be the finishing tool's entry point in the thread. A comparator makes an excellent tool for perfectly measuring the deviation. But most experienced setup people should be able to determine the deviation accurately enough with the internal measuring (sharp pointed) side of calipers.


Programming to allow for the deviation

The production program must, of course, deal with the deviation amount and polarity caused by two different threading rpms. For clarity, we first show an example program that uses Fanuc's G76 cycle to completely machine the thread with one tool at one rpm. This program probably resembles what you are currently doing. Note that we're only showing the threading operation. We're also assuming that a threading tool that forms the crest – a cresting insert – is being used (requiring a zero in-feed angle).

```
O0001 (Program machines entire
thread with one tool at one rpm)
.
.
(Rough and finish turning)
.
.
N200 M01 (Optional stop)
N205 T0505 M41 (Index to threading
tool, select low spindle range)
N210 G97 S500 M03 (Start spindle –
machining entire thread at 500 rpm)
N215 G00 X3.2 Z0.2 (Rapid to ap-
proach position)
N220 G76 X2.9125 Z-0.95 D0150 A0
F0.0625 (Machine entire thread)
N225 G00 X7.0 Z7.0 (Rapid to tool
change position)
N230 M30 (End of program)
```

Again, this program machines the entire thread with one tool at one rpm. Next we are going to show a program that machines the thread with one tool (again a crest forming tool requiring zero in-feed) at two rpms. Roughing will be done at 500 rpm and finishing will be done at 650 rpm.

During setup, the setup person runs the test program as described



Two new computer based training courses!

❖ **Setup Reduction For CNC**
❖ **Cycle Time Reduction For CNC**

Affordable courseware helps you improve productivity!

Setup & cycle time defined
During any CNC machine's usage, there are really only two activities. Either machines are in setup or they are running production. Setup time is the total time the machine is down between production runs. Cycle time is the time it takes to complete a production run divided by the number of good workpieces produced.

Setup reduction CD-rom course
When machines are in setup, they are not producing. Anything you can do to reduce setup time will improve the productivity of your CNC machines. In this **3 hour, 47 minute course**, we'll expose many techniques to help you reduce setup time and cost for the two most popular types of CNC machine tools.

We begin by presenting preliminary information, including justification issues and resources you have available to reduce setup time. We then present principles of setup reduction. We show two task types related to setup, three general ways to reduce setup time, and four steps to implementing setup reduction. Finally, we show countless specific techniques you can apply to reduce setup time in the approximate order that setups are made. 135 page manual is included

Cycle time reduction CD-rom course
During production runs, machines are supposed to be producing. Yet there may be activities occurring that are not very productive. In this **six hour course**, we'll show many techniques that will help you keep machines in cycle for as great a percentage of time as possible.

We begin by presenting preliminary information that will help you understand how to improve machine utilization. We then present principles of cycle time reduction, including the two tasks types related to running production, the three ways to reduce cycle time, and the four steps to reducing cycle time. Finally, we show countless specific techniques you can apply to reduce cycle time in the approximate order that production runs are completed. 140 page manual is included.

For CNC machining & turning centers
These courses address the two most popular forms of metal-cutting CNC machine tools - machining centers and turning centers. When appropriate, we separate presentations for the purpose of clarity. However, since you may be interested in only one machine type or the other, a few presentations during workpiece sizing and dull tool replacement are duplicated for both machine types.

Pricing:
CDR-STR ~ Setup reduction CD-rom course..... \$239.00
CDR-CTR ~ Cycle time reduction CD-rom course..... \$239.00



earlier. Say they determine that the 650 rpm scratch is to the left (negative side) of the 500 rpm scratch by 0.007 inch. This means that prior to the finishing pass/es, the starting position of the threading tool must be altered by 0.007 inch in the *positive Z* direction. This will ensure that the finishing pass/es will enter the thread at the same point as the roughing passes. Here is a modified program that shows the technique.

O0002 (Program machines thread with one tool using one rpm for roughing and another rpm for finishing)

(Rough and finish turning)

N200 M01 (Optional stop)

N205 T0505 M41 (Index to threading tool, select low spindle range)

N210 G97 S500 M03 (Start spindle – rough thread at 500 rpm)

N215 G00 X3.2 Z0.2 (Rapid to approach position)

N220 G76 X2.9185 Z-0.95 D0150 A0 F0.0625 (Rough thread within 0.006 of finished diameter – 0.003 is left on the side)

N225 G00 W0.007 S650 (Move over in Z by positive 0.007 inch, select finishing speed)

N230 G92 X2.9165 Z-0.95 F0.0625 (Make first finishing pass – 0.001 deep)

N235 X2.9145 (Make second finishing pass – 0.001 deep)

N240 X2.9135 (Make third finishing pass – 0.0005 deep)

N245 X2.9125 (Make fourth finishing pass – 0.0005 deep)

N250 X2.9125 (Make free-ride pass – 0.0 deep)

N255 G00 X7.0 Z7.0 (Rapid to tool change position)

N260 M30 (End of program)

In line N220, we're rough machining the thread at 500 rpm to within 0.006 of the finished threading diameter (0.003 material is still left on the side). In line N225, we're making an incremental movement in Z equal to the amount of deviation determined during setup.

Since the setup person found that the second witness mark line was to the left (negative side) of the first witness mark line, the movement must be in the plus direction to counter the deviation caused by rpm changing. The motion is being commanded by a W word (Fanuc's word for incremental Z motion). If you'd rather stay in the absolute mode, you can, of course, replace the W0.007 with Z0.207. Since it is a setup person that must manipulate this word, make it as easy as possible for them to do so. We feel the W word makes it about as easy as it gets.

If, of course, the second witness mark line is to the right of the first, the W word must be negative to counter the effect of the rpm change.

How often do you have to test?

If you are using the same tool to rough and finish (as the previous example shows), the deviation caused by rpm changing will remain consistent from one time you run a job to the next. This means that once the setup person has determined the amount of deviation and changed the program once, it will remain correct every time the job is run.

However, the amount of deviation will only remain consistent for one set of rpm changes. If your setup person changes the finishing speed to 700 rpm (instead of 650 rpm), they must also repeat the test and modify the production program accordingly. And of course, this technique will have to be used when machining threads on other workpieces of different diameters (requiring different rpms).

What about other in-feed angles?

Our example assumed the use of a cresting insert that requires a zero in-feed angle (A0 in the G76 command). If you're not using a cresting insert, things get quite a bit tougher.

With a non-cresting insert, most programmers want the threading tool to maintain a 30 degree in-feed angle that causes the tool to machine on only the front side of the threading insert. This is easily programmed with Fanuc's G76 command by including the word A60 in the G76 command (it's equally easy with other controls as well using a similar technique).

You must understand what happens when you include A60 in the G76 command. For each successive pass, the control actually alters the Z axis starting point for the threading tool. The starting point is altered based upon the current threading depth. For the first pass, the control will not alter the Z axis starting position. But say the second pass will be 0.010 deep. The control will alter the Z axis starting point in the negative Z direction by 0.0057 (the tangent of 30 degrees times the 0.010 depth).



New computer based training course! CD-ROM disk: over 6 hours!

CNC Machining Center Programming, Setup, and Operation

Affordable courseware for CNC machining centers!

A very popular CNC machine type!

CNC machining centers are among the most popular types of CNC machine tools. Most companies that have any CNC machines have at least one. Unfortunately, companies are finding it more and more difficult to find and hire qualified CNC people. Many are realizing that they must provide extensive training to new hires and provide at least some continuing training to established employees.

A proven method

This affordable courseware makes it possible to train CNC people from scratch. While we assume the student has some basic machining practice experience, we assume nothing about their previous CNC skills. Using our proven *key concepts approach*, we bring students up to speed gradually – constantly building upon previously presented information – and we stress the reasons *why* things are done as importantly as *how* they're done. Six of the ten key concepts are most related to programming, and four are related to setup and operation.

24 lessons!

We further divide the key concepts into twenty-four lessons. Lessons range from under five minutes to just over twenty minutes in length (total course presentation time is just over six hours on one CD-rom).

The most popular control!

All examples are shown in the format for the most popular control in the industry – the Fanuc control. Note that many control manufacturers claim to be Fanuc-compatible.

What you get!

The CD-rom disk is jam-packed with over six hours of information about CNC machining centers. By itself, it makes formidable training tool. It's price is \$149.00.

When purchased with the workbook answer combination (\$30.00), you'll be able to confirm that you truly understand the material (24 exercises, one for each lesson).

When purchased with the optional student manual (\$60.00), you'll have a way to easily review information after you finish the course.

If all items are purchased (total: \$239.00), we include a one-year subscription to our newsletter, The Optional Stop.



The price for unlimited training? ... \$149.00 (courseware only)

Companion manual: \$60.00, workbook/answer combination: \$30.00

Contact CNC Concepts, Inc. (847-639-8847) to order!

This means that for each successive threading pass, the tool will start at a different Z starting position.

This makes it more difficult to determine the tool's correct starting position after the roughing operation is completed. To eliminate the problem, you may opt to start using cresting inserts. But if cresting inserts are not an option, you must be able to determine the tool's starting Z position for its last roughing pass. We offer two ways to do so.

The first (and the easiest) is to monitor the Z axis position display during the G76 cycle. As the tool begins its X axis approach movement for the last pass, stop the cycle (which you can do if dry run is turned on) and write down the Z axis position display value. This will be the point of reference for manipulating the starting position for the finishing passes.

Another way to determine the threading tool's last Z position is to calculate it. From the tool's initial approach position, subtract this calculated value:

Tangent of half the tool angle times the result of the total thread depth minus the first pass depth

For our example (a 0.0625 pitch thread having a 0.0437 depth), the first pass depth was 0.015 (D0150 in line N220 of the most recent program

shown). Subtract this value from 0.0437 and you get 0.0287. The tangent of 30 degrees is 0.5773. When multiplied times 0.0287, the result is 0.0165. The tool's last pass Z axis approach position should be 0.1835 (0.2 initial approach position minus 0.0165). This will be the point of reference for determining the finish pass (altered) approach position.

Admittedly, this gets a little complicated – and again – you can eliminate the problem by using cresting inserts and zero in-feed. But regardless of how difficult it seems to you, remember the benefits of roughing and finishing at two speeds – especially for long production runs. And as stated, once the setup person has correctly adjusted the program, it will remain correct every time the job is run. It will be well worth your time to stick with it and figure out what it takes to make it work!

What about different thread forms?

Our example described the use of a 60 degree threading tool. All major points apply to other thread forms. Consider, for example, an ACME thread having an included angle of 29 degrees. While the witness mark left after bluing may not be as pronounced, the setup person will still be able to determine the amount and


polarity of deviation. And if a non-cresting insert is used (commonly the case with ACME threads), you'll have to use one of the techniques we've provided for determining the point of reference after the roughing passes have been made (last Z axis approach position). If calculating, the tangent of 14.5 degrees (half of 29) must be used instead of the tangent of 30 degrees.

Rough and finish threading with two tools

About the only new point that must be made about rough and finish threading with two tools has to do with the fact that testing must be done every time the job is run (assuming the rough and finish threading tools have been removed from the turret since the last time the job was run). Since this is the case, you'll want to make it as easy as possible for the setup person to perform the test and manipulate the programs. Here is an example program that includes trial machining. For our example, the trial machining commands are included in a sub-program (they could be included in your main program). The trial machining command begins with a slash code and will only be executed if the block delete switch (also called the optional block skip switch) is turned off. During normal production, the block delete switch will be kept on.

Note that if you desire, you can use slashed-coded trial machining commands in your programs that rough and finish with the same threading tool, but since testing must only be done one time, they may not be as needed.

Again, we're machining the national standard 3"-16 thread with a crest forming tool requiring no in-feed. Tool number five is the roughing tool and tool number six is the finishing tool.



New computer based training course! CD-ROM disk: 5.8 hours!
CNC Turning Center Programming, Setup, and Operation
Affordable courseware for CNC turning centers!
28 lessons!

A very popular CNC machine type!
CNC turning centers are among the most popular types of CNC machine tools. Most companies that have any CNC machines have at least one. Unfortunately, companies are finding it more and more difficult to find and hire qualified CNC people. Many are realizing that they must provide extensive training to new hires and provide at least some continuing training to established employees.


A proven method
This affordable courseware makes it possible to train CNC people from scratch. While we assume the student has some basic machining practice experience, we assume nothing about their previous CNC skills. Using our proven *key concepts approach*, we bring students up to speed gradually – constantly building upon previously presented information – and we stress the reasons *why* things are done as importantly as *how* they're done. Six of the ten key concepts are most related to programming, and four are related to setup and operation.

The most popular control!
All examples are shown in the format for the most popular control in the industry – the Fanuc control. Note that many control manufacturers claim to be Fanuc-compatible.

What you get!
The CD-rom disk is jam-packed with 5 hours, 52 minutes of information about CNC turning centers. By itself, it makes formidable training tool. It's price is \$149.00. When purchased with the workbook answer combination (\$30.00), you'll be able to confirm that you truly understand the material (28 exercises, one for each lesson). When purchased with the optional student manual (\$60.00), you'll have a way to easily review information after you finish the course.

If all items are purchased, we include a one-year subscription to our newsletter, The Optional Stop.

The price for unlimited training? ... \$149.00 (courseware only)
Companion manual: \$60.00, workbook/answer combination: \$30.00



O0003 (Program machines thread with two tools using one tool and rpm for roughing and another tool and rpm for finishing)

(Rough and finish turning)

N195 M01 (Optional stop)

/N200 M98 P2000 (If block delete switch is off, execute trial machining program)

N205 T0505 M41 (Index to rough threading tool, select low spindle range)

N210 G97 S500 M03 (Start spindle – rough thread at 500 rpm)

N215 G00 X3.2 Z0.2 (Rapid to approach position)

N220 G76 X2.9185 Z-0.95 D0150 A0 F0.0625 (Rough thread within 0.006 of finished diameter – 0.003 is left on the side)

N225 G00 X7.0 Z7.0 (Rapid to tool change position)

N230 M01 (Optional stop)

N235 T0606 M41 (Index to finish threading tool, select low spindle range)

N240 G97 S650 M03 (Start spindle – finish at 650 rpm)

N245 G00 X3.2 Z0.2 (Rapid to approach position)

N250 G00 W0.007 (THIS VALUE MUST BE CHANGED AFTER TEST!)

N255 G92 X2.9165 Z-0.95 F0.0625 (Make first finishing pass – 0.001 deep)

N260 X2.9145 (Make second finishing pass – 0.001 deep)

N265 X2.9135 (Make third finishing pass – 0.0005 deep)

N270 X2.9125 (Make fourth finishing pass – 0.0005 deep)

N275 X2.9125 (Make free-ride pass – 0.0 deep)

N280 G00 X7.0 Z7.0 (Rapid to tool change position)

N285 M30 (End of program)

Notice that we've included the trial machining command (line N200) just prior to threading. At this point in the program, the rough and finish turning operations will have been completed and the diameter should be at 3.0000 inches and ready to thread.

You've probably noticed that the actual threading commands are much the same as in the previous example. We've simply commanded a tool change prior to finishing. Additionally, notice the bold message in line N250. This should make quite

clear to the setup person which command must be changed after testing. And again, we're using the simple incremental method (W word) to make the deviation amount as easy as possible to enter.

Here is the trial machining sub-program. Remember, it will only be executed when the block delete switch is off (during setup). We've also included six tries per tool. After each try, the setup person can check to see whether the tool has left a witness mark. If it has not, they simply press cycle start again. The tool's offset will be reduced by 0.002 inch (0.001 on the side), the tool will make another pass, and stop again. If the tool has cut a line, the setup person turns ON the block delete switch to skip the balance of the passes. Note that between the roughing tool and the finishing tool, the setup person must turn the block delete switch back off. If the tool has not made a witness mark line after six tries, the message next to the program stop will tell the operator to re-run the threading portion of the program (the tool was not within 0.010 of the workpiece at the start).

O2000 (Trial machining sub-program)

T0505 M41 (Index to rough threading tool, select low spindle range)

G97 S500 M03 (Start spindle at roughing speed)

G00 X3.2 Z0.2 (Rapid to approach position)

G92 X2.998 Z-0.95 F0.0625 (Make one very shallow pass – just scratch the workpiece)

M00 (Did the tool make a line? – If so, turn ON block delete)

/G10 P5 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)

/T0505 (Re-instate offset)

/G92 X2.998 Z-0.95 F0.0625 (If necessary, make another pass)

/M00 (Did the tool make a line? – If so, turn ON block delete)

/G10 P5 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)

/T0505 (Re-instate offset)

/G92 X2.998 Z-0.95 F0.0625 (If necessary, make another pass)

/M00 (Did the tool make a line? – If so, turn ON block delete)

/G10 P5 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)

/T0505 (Re-instate offset)

/G92 X2.998 Z-0.95 F0.0625 (If necessary, make another pass)

/M00 (Did the tool make a line? – If so, turn ON block delete)

/G10 P5 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)

/T0505 (Re-instate offset)

/G92 X2.998 Z-0.95 F0.0625 (If necessary, make another pass)

/M00 (Did the tool make a line? – If so, turn ON block delete)

/G10 P5 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)

/T0505 (Re-instate offset)

/G92 X2.998 Z-0.95 F0.0625 (If necessary, make another pass)

/M00 (If no witness mark line at this point, the tool was not within 0.01 of the workpiece at start – you must re-run the threading portion of the program from the beginning)

G00 X7.0 Z7.0 (Rapid to tool change position)

M00 (BLUE-UP THE WORKPIECE AND TURN OFF BLOCK DELETE)

T0606 M41 (Index to finish threading tool, select low range)

G97 S650 M03 (Start spindle at finishing rpm)

G00 X3.2 Z0.2 (Rapid to approach position)

N4 G92 X2.998 Z-0.95 F0.0625 (Make one very shallow pass – just scratch the workpiece)

M00 (Did the tool make a line? – If so, turn ON block delete)

/G10 P5 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)

/T0606 (Re-instate offset)

/G92 X2.998 Z-0.95 F0.0625 (If necessary, make another pass)

/M00 (Did the tool make a line? – If so, turn ON block delete)

/G10 P5 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)

/T0606 (Re-instate offset)

/G92 X2.998 Z-0.95 F0.0625 (If necessary, make another pass)

/M00 (Did the tool make a line? – If so, turn ON block delete)

/G10 P5 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)

/T0606 (Re-instate offset)

/G92 X2.998 Z-0.95 F0.0625 (If necessary, make another pass)

/M00 (Did the tool make a line? – If so, turn ON block delete)

/G10 P5 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)

/T0606 (Re-instate offset)

/G92 X2.998 Z-0.95 F0.0625 (If necessary, make another pass)

/M00 (Did the tool make a line? – If so, turn ON block delete)

/G10 P5 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)

/T0606 (Re-instate offset)

```

/G92 X2.998 Z-0.95 F0.0625 (If necessary, make another pass)
/M00 (If no witness mark line, the tool was not within 0.01 of the workpiece at start – you must rerun the threading portion of the program from the beginning)
G00 X7.0 Z7.0 (Rapid to tool change position)
M00 (DETERMINE THE DEVIATION AMOUNT AND DIRECTION AND CHANGE THE W WORD IN THE MAIN PROGRAM)
M99 (End of sub-program)

```

Does your machine have custom macro B?

Admittedly, the previous example may look a little complicated. But remember, we're trying to simplify things for the setup person (not the programmer). And truly, this technique makes it about as simple for the setup person as it gets. But if you need to use this technique often (many different workpieces to thread), you'll have to write a lot of tedious commands. If your machine has custom macro B, however, you can write one universal program (to replace the sub-program) that will work for any thread of any diameter with any pitch! And the custom macro will be shorter and (we think) easier to follow. Here's the revised example main program:

```

O0003 (Program machines thread with two tools using one tool and rpm for roughing and another tool and rpm for finishing)
.
.
(Rough and finish turning)
.
.
N195 M01 (Optional stop)
/N200 G65 P2001 X3.000 Z0 W0.95 R5.0 F6.0 A500.0 B650.0 C0.0625 (If block delete switch is off, execute trial machining custom macro program)
N205 T0505 M41 (Index to rough threading tool, select low spindle range)
N210 G97 S500 M03 (Start spindle – rough thread at 500 rpm)
N215 G00 X3.2 Z0.2 (Rapid to approach position)
N220 G76 X2.9185 Z-0.95 D0150 A0 F0.0625 (Rough thread within 0.006 of finished diameter – 0.003 is left on the side)
N225 G00 X7.0 Z7.0 (Rapid to tool change position)
N230 M01 (Optional stop)

```

```

N235 T0606 M41 (Index to finish threading tool, select low spindle range)
N240 G97 S650 M03 (Start spindle – finish at 650 rpm)
N245 G00 X3.2 Z0.2 (Rapid to approach position)
N250 G00 W0.007 (THIS VALUE MUST BE CHANGED AFTER TEST!)
N255 G92 X2.9165 Z-0.95 F0.0625 (Make first finishing pass – 0.001 deep)
N260 X2.9145 (Make second finishing pass – 0.001 deep)
N265 X2.9135 (Make third finishing pass – 0.0005 deep)
N270 X2.9125 (Make fourth finishing pass – 0.0005 deep)
N275 X2.9125 (Make free-ride pass – 0.0 deep)
N280 G00 X7.0 Z7.0 (Rapid to tool change position)
N285 M30 (End of program)

```

Only line N200 has changed. We're passing some variables to the custom macro to tell the custom macro about the current thread to machine. X (#24) is the diameter into which the thread will be machined. Z (#26) is the position in Z at which the thread starts. W (#23) is the distance in Z we want the threading tool to move. R (#18) specifies the rough threading tool's turret station number and F (#9) specifies the finish threading tool's turret station number. A (#1) specifies the speed for roughing and B (#2) specifies the speed for finishing. And C (#3) specifies the thread's pitch.

Here is the custom macro:

```

O2001 (Custom macro for test cutting threads with two tools)
T[#18*100+#18] M41 (Index to rough threading tool, select spindle range)
G97 S#1 M03 (Start spindle at roughing speed)
G00 X[#24+0.2] Z[#26+0.2] (Rapid to approach position)
N1 G92 X[#24-0.002] Z[#26-#23] F#3 (Make one very shallow pass – just scratch the workpiece)
#3006 = 101 (LINE? TURN ON BLOCK DELETE)
/G10 P#18 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)
/T[#18*100+#18] (Re-instate offset)
/GOTO 1 (Make another pass)
G00 X7.0 Z7.0 (Rapid to tool change position)
#3006 = 102 (ADD BLUE - TURN OFF BLOCK DELETE)
T[#9*100+#9] M41 (Index to finish threading tool, select low range)

```

```

G97 S#2 M03 (Start spindle at finishing rpm)
G00 X[#24+0.2] Z[#26+0.2] (Rapid to approach position)
N2 G92 X[#24-0.002] Z[#26-#23] F#3 (Make one very shallow pass – just scratch the workpiece)
#3006 = 103 (LINE? TURN ON BLOCK DELETE)
/G10 P#9 U-0.002 (Reduce offset by 0.002 – 0.001 on the side)
/T[#9*100+#9] (Re-instate offset)
/GOTO 2 (Make another pass)
G00 X7.0 Z7.0 (Rapid to tool change position)
#3000 = 100 (CHANGE W WORD IN MAIN PROGRAM)
M99 (End of sub-program)

```

Note that this program lets the setup person make as many tries as necessary until the threading tool makes the witness mark line. It puts the machine into a "loop" until the setup person eventually turns on the block delete switch to exit the loop (the tool has left the witness mark). However, the setup person must still remember to turn off the block delete switch after the roughing tool has made the witness mark. To make it as clear as possible, we've used the #3006 (stop with message commands) to force the message to be shown on the display screen instead of M00 commands.

Note that with this program (as well as the previous example) the setup person will have left the block delete switch on at the completion of the test, meaning that once they start running production, the switch will be in the correct position (skipping trial machining from now on).

Also, the setup person must still alter the main program to include the correct deviation and direction (line N250 in our main program). Since they must not continue running the program until this is done, we're using an alarm generating #3000 command at the end of the custom macro. Once the setup person has altered line N250, they'll simply run the main program from the beginning. Again, block delete should currently be on, and the control will skip any additional trial machining.

M01

The Optional Stop Newsletter Enclosed!

Mailroom: PAID SUBSCRIPTION ENCLOSED. Please assure delivery to addressee. Thank You!

Products That Address The Needs Of CNC Technology Today!

Use this handy form to request more information about our products.

CNC Video Courses:

- Machining Center Programming And Operation
- Turning Center Programming And Operation
- Conversational Turning Center Programming & Op
- Custom Macro Programming
- Four Axis Turning Center Programming

CD-rom Courses:

- Machining Center Programming, Setup, & Operation
- Turning Center Programming, Setup, & Operation
- Setup Reduction for CNC
- Cycle Time Reduction for CNC
- Parametric Programming
- CNC Router Programming, Setup, & Operation
- Advanced Techniques With Basic CNC Features

CNC Mini-Vids:

- Related To Selected CNC Topics Like Setup Time Reduction, Cycle Time Reduction, Probing

CNC Course Curriculumms For Instructors:

- Machining Center Programming And Operation
- Turning Center Programming And Operation
- CNC Setup Reduction for CNC
- CNC Router Programming And Operation
- Maximizing CNC Utilization (NEW!)

If other than above:

Name: _____

Company Name: _____

Address: _____

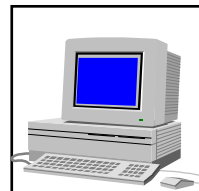
City, State, Zip: _____

CNC Publications:

- Machining Center Self Study Course
- Turning Center Self Study Course
- CNC Router Self Study Course
- Managing CNC Operations
- CNC For Machining
- CNC Advanced Techniques
- CNC Accessory Devices
- The Optional Stop Subscription & Back Issues

CNC In-Plant Training:

- Basic CNC Courses
- CNC Tune-Up - For your *experienced* People



Check Out Our Website!

<http://www.cncci.com/>

Information about each of our products is on line!